NASA Technical Memorandum 103293

A Three-Dimensional Turbulent Heat Transfer Analysis for Advanced Tubular Rocket Thrust Chambers

Kenneth J. Kacynski
Lewis Research Center
Cleveland, Ohio

Prepared for the
JANNAF Propulsion Conference
Anaheim, California, October 3–5, 1990

NASA

A THREE-DIMENSIONAL TURBULENT HEAT TRANSFER ANALYSIS FOR ADVANCED TUBULAR ROCKET THRUST CHAMBERS

Kenneth J. Kacynski
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135

ABSTRACT

Heat transfer was analyzed in the throat region of a plug and spool rocket engine for both smooth and corrugated walls. A three-dimensional, Navier-Stokes code was used for the analysis. The turbulence model in the code was modified to handle turbulence suppression in the crevice region of the corrugated wall. Circumferential variations in the wall heat transfer were predicted for the corrugated wall. The overall heat transfer at the throat for the corrugated wall was 34 percent higher than it was for the smooth wall for comparable rocket flow conditions.

INTRODUCTION

Estimation of wall heat transfer is critical in the design and optimization of high pressure, high heat flux, tubular coolant chamber rocket engines. Unfortunately, very little work has been previously done to estimate heat flux variations in the crown/crevice regions of corrugated rocket engine walls. Currently, most rocket engine designers assume that heat flux rates in crevice regions are identical to the crown regions or assume that the corrugated surface acts globally like a smooth wall, with the same amount of total heat transfer. In the past, such simplifying assumptions were acceptable. Previously, corrugated rocket wall surfaces were associated with high temperature, low thermal conductivity materials that only had application to low heat flux regions. However, Kazaroff and Pavli (Ref. 1) have found that coolant chambers constructed of copper may have significant advantages, such as increased life and chamber pressure, over milled coolant channels for high heat flux applications. In this regime, it becomes imperative, from both a design and a system integration perspective, to have better knowledge of the heat transfer characteristics of tubular coolant chambers. This paper analytically addresses the hot side heat transfer aspects of tubular chambers and compares heat transfer rates of a tubular wall to a smooth wall with otherwise similar geometry and boundary conditions.

As the corrugated wall rocket engine is a three-dimensional configuration, conventional rocket engine computer predictions, such as the JANNAF (Joint Army, Navy, NASA, Air Force) recommended TDK/BLM procedure (Ref. 2), using smooth wall assumptions for the calculation of heat transfer rates, are not applicable. Since significant variations in velocity and temperature around the periphery of the tube (corrugated wall) will exist, the ability to capture this effect dictates that a three-diminsional computer analysis be performed. For this study, heat transfer predictions were made with a three-dimensional (cartesian coordinates) Navier-Stokes code, PARC3D (Ref. 3). A hydrogen-oxygen rocket, with an extensive smooth wall experimental database, was analyzed in this study.

HARDWARE DESCRIPTION

The rocket engine analyzed in this study was the plug-and-spool configuration (Fig. 1) that has been tested at NASA Lewis Research Center (Ref. 4). Due to the simplicity of manufacturing, the spoolpiece portion of this configuration has served as the test specimen while the plug has been used as a means of producing sonic flow conditions. To date, extensive smooth wall fatigue tests have been performed and current plans are to perform similar tests with corrugated spoolpiece rocket engines (Ref. 1).

Smooth walled test articles have been made by milling 72 coolant passages into a cylindrical, roughly bored out, copper spoolpiece. This was followed by electroform closeout of the passages and then the finishing boring cuts of the interior surface (the hot gas side) were made.

In maintaining similarity, the work being performed by Kazaroff and Pavli involves the electroform bonding of 72 tubes, each tube having an outside diameter of approximately one-eighth of an inch, to form a rocket engine spoolpiece test article. The interior (i.e., the hot gas side) of the tubular spoolpiece rocket has the appearacnce of a corrugated wall and all discussion of corrugated geometries described in this study refer to the hot gas side surface created in constructing a rocket engine out of tubes.

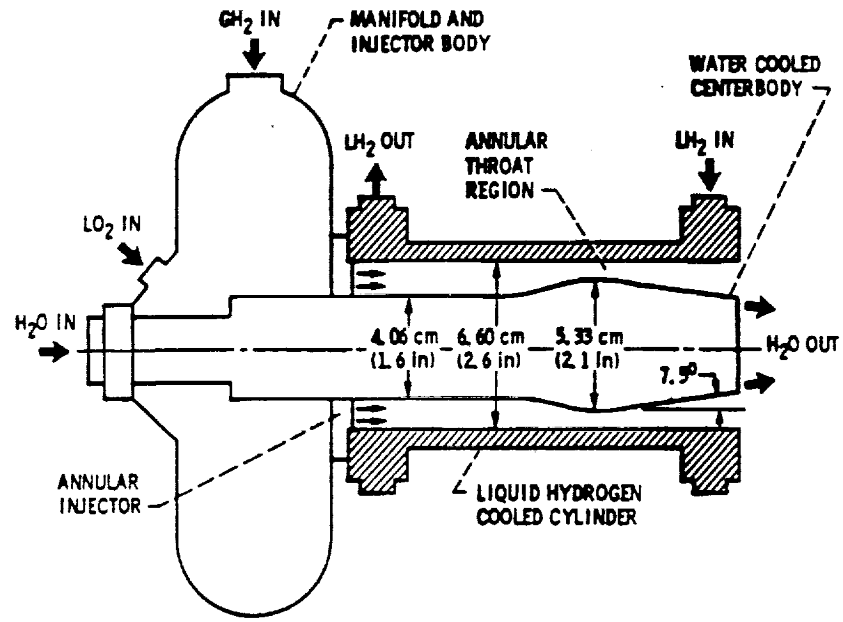

Figure 1. - Schematic of cylindrical thrust chamber assembly.

ANALYSIS

The computer code, PARC3D, was used for this study. This code is a compressible, three-dimensional, spatially second-order accurate, Navier-Stokes solver. The equations of mass, momentum, and energy, are solved in a factored, implicit manner. A constant specific heat ratio was assumed. The goal of this study was to perform a comparative evaluation of heat transfer between corrugated and smooth walls, therefore chemical recombination and variable specific heats were not used. It was felt that they would increase computational time with little gain in solution accuracy.

An algebraic turbulence model, similar in nature to the Baldwin-Lomax Model, was employed for the calculation of the turbulent viscosity. The model is discussed in a later section. The fluid viscosity was assumed to obey the Sutherland Viscosity Law (Ref. 5).

The input necessary to simulate the rocket engine included a full geometry description, Reynolds number, laminar and turbulent Prandtl number (both assumed constant), and appropriate boundary conditions. The values of Reynolds and Prandtl Numbers used in the calculations represent the chamber conditions in a hydrogen-oxygen rocket engine operating at a mixture ratio of 6. The turbulent Pandtl was set to 0.9. Additional input requirements specific to the PARC3D Code were values of artificial diffusion (both second and fourth order) and time step limitations. Default values were used for both types of artificial diffusion. Local time-step advancement was applied to enhance acceleration of the solution to steady-state.

A steady-state solution of the Navier-Stokes equations was attained when the parameters of interest, the velocity and temperature near the wall, converged. Convergence of these parameters occurred in about 30 000 time steps. This is substantially (an order of magnitude) greater than the number of iterations required to resolve freestream pressure and temperatures and therefore, the resolution of near-wall velocity and temperature was the appropriate convergence criteria for this rocket engine analysis.

BOUNDARY CONDITIONS

Appropriate boundary conditions for the nozzle inlet, exit, and walls were required. At the nozzle inlet, total pressure and temperature were specified. These conditions corresponded to a hydrogen-oxygen rocket engine operating at an O/F of 6 and a chamber pressure of 600 psia. At the nozzle exit, no boundary conditions were required in the supersonic region. In the subsonic region, the exit static pressure was required. This exit pressure was estimated by first analyzing an inviscid nozzle flow. As this case has a fully supersonic exit, specification of static exit pressure was unnecessary. The static pressure at the wall in the exit plane from the inviscid solution was then used as the static pressure input to the fully viscous case. At the spoolpiece wall, no-slip, isothermal (1400 °R) boundary conditions were imposed. At the plug wall, slip boundary conditions were used.

GRID GENERATION

Algebraic relationships were used to generate the grids for both the smooth and corrugated plug-and-spool configurations. As illustrated in Fig. 2, a significant amount of grid packing was used to accurately determine conditions at the throat wall, especially in the crevice region. As illustrated in Fig. 2, a three-dimensional computation was done for both the smooth walled and the corrugated walled engine. This was done to maintain numerical similarities between the smooth and corrugated wall engines and also allowed examination of any anomolies that may have resulted from excessive grid skewness or artificial diffusion. No significant anomolies, however, were observed.

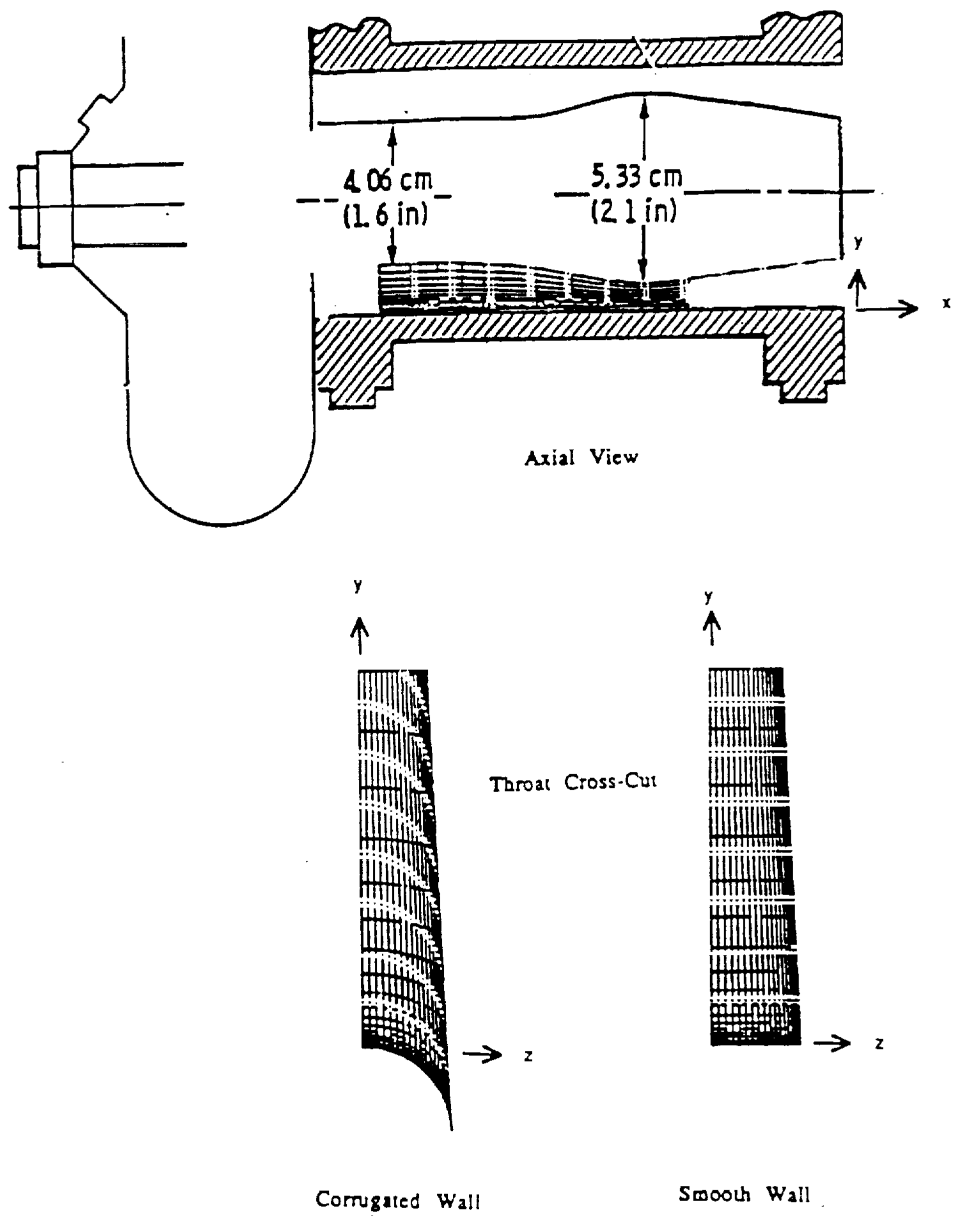

Figure 2. Computational Grid (30 X 30 X 20) Generated for Smooth/Corrugated Wall Rocket Engine Comparison

TURBULENCE MODELING

The turbulence model in the PARC3D Code is of an algebraic type similar to the Baldwin-Lomax formulation outlined in Ref. 6. The application of this turbulence model was not acceptable for the corrugated wall analysis. There were two difficulties in using the turbulence formulation. Both difficulties arise in establishing the correct mixing length in the inner region of the boundary layer. One problem encountered was that the PARC3D Code estimates distances from a solid wall to a point in space by assuming that there are no significant curvatures within the inner region of the boundary layer. In cases of significant curvatures, such as the corrugated wall configuration, substantial errors would occur with this formulation. This problem was alleviated by slightly modifying the code to correctly predict the normal distance from a grid point to a curved wall.

The other difficulty of using algebraic modelling for the turbulence is that turbulence suppression will likely occur in the crevice region between tubes that further dampens the turbulence. The imposition of symmetry boundary conditions at the cross stream boundary of the computational domain precluded any possibility that the PARC3D Code could handle this situation. The dampening of the turbulence in the crevice region was addressed by applying a mixing length formulation similar to one used by Kadle and Sparrow (Ref. 7) for the calculation of turbulence near corners in fully developed duct flow. The formulation used for this analysis employed the following relation for the calculation of the inner layer mixing length in the crevice regions;

$$l = \frac{1}{\sqrt{1/l_{n1}^2 + 1/l_{n2}^2}}$$

where l_{n1} and l_{n2} are the normal distances from the tube walls to the grid point in question, see Fig. 3.

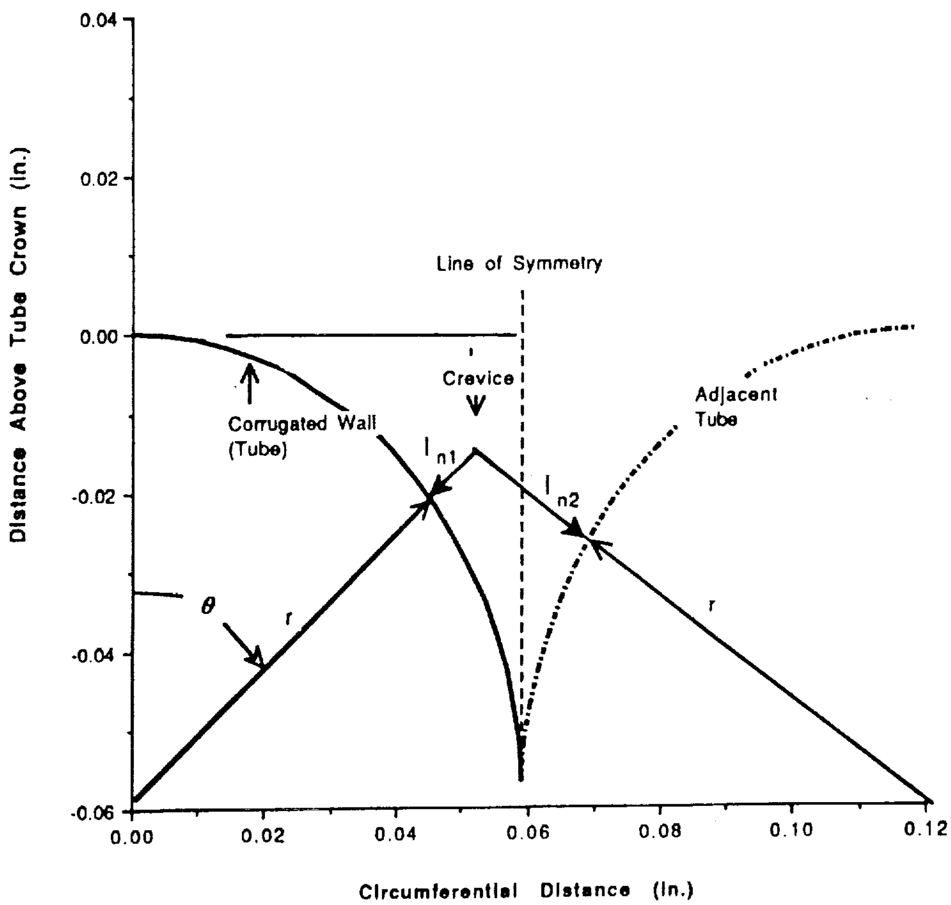

Fig. 3 Turbulence Mixing Length Parameters

HEAT TRANSFER PREDICTIONS

Rocket engine heat transfer is a critial parameter in evaluating an engine design, as it effects system performance and engine life. Unfortunately, the estimation apriori of an engines heat transfer characteristics is a very difficult task and heavy reliance on experimental data is necessary. Therefore, analytical predictions of heat transfer are best preformed applying both analysis and experimental data. The corrugated/smooth wall comparison is ideally suited for such a combination of empericism and analysis. This is because the geometries are fairly similar and an extensive heat transfer database for the smooth walled configuration already exists. Consequently, expected experimental heat transfer rates from a corrugated wall rocket engine can be determined by employing the following relation:

$$\left[q''_c\right]_E = \left[q''_s\right]_E \left[\frac{q''_c}{q''_s}\right]_A$$

The subscripts c and s represent conditions at the corrugated and smooth wall, respectively. The subscripts A and E refer to analytic predictions and experimentally measured results, respectively. Previous experimental measurements (Ref. 5) indicate that the smooth wall heat flux q''_s, at the throat of the plug and spool configuration is 35 Btu/in^2-sec. The results presented in the next section will show the variation of the heat flux ratio, q''_c/q''_s, determined analytically.

RESULTS

Figure 4 illustrates the variation of the heat ratio, q''_c/q''_s, as a function of the wall angle. It is seen that heat fluxes decay appreciably in regions very near the crevice ($\theta \geq 60°$). This decay occurs for two reasons. First, the production of turbulence is suppressed in these regions. Second, at the half-tube boundary it is required that conditions normal to this boundary be symmetrical. As the boundary is also very close (and nearly parallel) to the no-slip, isothermal boundary conditions imposed on the corrugated wall, increasingly isothermal conditions (hence no heat transfer) will be approached as the two boundaries become closer, and more parallel, to each other. Where the two boundaries meet (in the corner formed by joining two tubes) no temperature gradient can exist. At this point, the heat transfer rate, by necessity, must be zero. However, the mixing ability of turbulent flow serves to localize the severe extent of this heat transfer suppression as evidenced by the fact that the first boundary point away from the crevice has a heat transfer decay, compared to a smooth walled geometry, of only 22 percent.

Total heat transfer rates at the throat can be determined by integration of the heat fluxes around the periphery of the smooth wall and the corrugated wall:

$$\frac{Q'_c}{Q'_s} = \frac{r\int q''_c d\theta}{q''_s\int dr} = \frac{\int q''_c d\theta}{q''_s}$$

Numerical integration of the above relation indicates that total heat transfer (at the throat) will be 34 percent greater for the corrugated wall than the smooth wall. This enhancement of heat transfer is attained primarily from the increased surface area (53 percent) of the corrugated wall, compared to the smooth wall.

Fin effectiveness, defined as the ratio of the fin heat transfer rate to the heat transfer rate that would exist without the fin, is an important term in heat exchanger design (Ref. 8). For the corrugated wall, which behaves like a fin, the effectiveness is thus 1.34 at the throat. Further insight into the local significance of fin effectiveness can be realized by examination of the local fin effectiveness term, $q_c r d\theta/q_s dr$. This term represents the normal projection of the corrugated wall heat flux onto an equivalent smooth walled surface. The resultant circumferential variation of the local fin effectiveness parameter is illustrated in Fig. 5 for the plug and spool geometry. From this figure, it becomes apparent that there is an insignificant amount of heat transfer enhancement near the crown of the corrugated wall as the local fin effectiveness is very close to 1. However, in the crevice regions, the local fin effectiveness approaches a value of 5. The large variation of local fin effectiveness between the crown and the crevice region is a very significant observation that will be discussed further in the recommendations section. It should be noted that right at the crevice, the numerator and denominator of the local fin effectiveness equation are both zero and the term becomes indeterminate. Fortunately, only a very small region, dependant only on the numerical resolution of the crevice area, is affected by this anomoly.

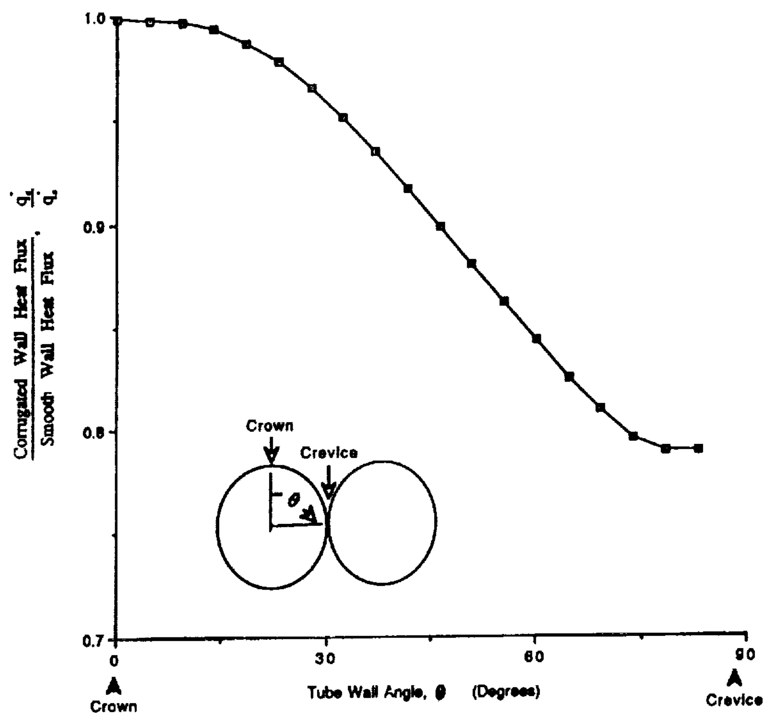

Figure 4. Heat Flux Comparison between Smooth and Corrugated Wall Rocket Engines

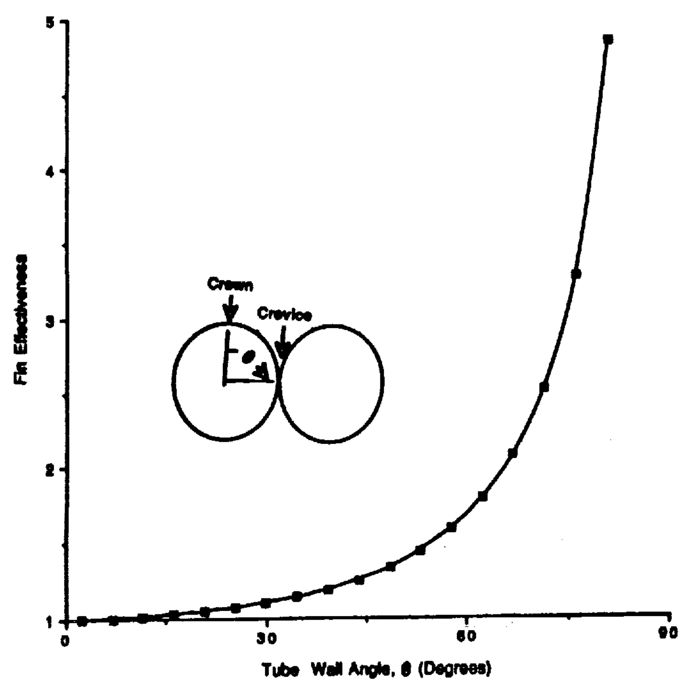

Figure 5. Local Variation of the Fin Effectiveness of Corrugated Wall Rockets

RECOMMENDATIONS AND CONCLUSIONS

The results of the corrugated/smooth wall analysis show that substantial variations in heat transfer rates can be expected around the periphery of a corrugated wall. Compared to a smooth wall, a net increase in heat transfer can be expected because the increased surface area of the corrugated geometry more than compensates for any decrease of heat transfer rates that may occur in the crevice region. From a heat transfer/engine performance perspective, the result is highly desirable as total heat transfer to the coolant can be increased without the need to increase local heat fluxes or increase engine size, both options of which have undesirable side effects (higher coolant pressure drop and engine weight, respectively).

Another important observation of this analysis is that most of the heat transfer enhancement is occurring in the crevice regions of the corrugated wall. In actual practice, the presence of this very steep crevice region is very dependent on the method of constructing a tubular rocket engine. If the engine is constructed as described by Kazaroff and Pavli (i.e., elecroform bonded) then a significant crevice region will exist. If, instead, the tubes were to be bonded by a brazing process that fills the crevice region, little enhancement of heat transfer would occur.

Additional efforts in this area should be directed towards obtaining experimental data and advancing the current JANNAF methodology of predicting rocket engine performance. As tubular and other three-dimensional configurations are further applied in rocket engine designs, there is an increasing need for accurate and efficient analytical methods that can be applied to these configurations.

REFERENCES

1. Kazaroff, J.M. and Pavli, A.J. "Advanced Tube-Bundle Rocket Thrust Chambers," AIAA Paper No. 90-2726, 26th Joint Propulsion Conference, July 1990.

2. Nickerson, G.R., et al.: Engineering and Programming Manual: Two-Dimensional Kinetics (TDK) Nozzle Performance Computer Program. (SN91, Software and Engineering Associates; NASA Contract NAS8-36863) NASA-CR-890124, 1989.

3. Cooper, G.K.: The PARC Code: Theory and Usage. AEDC-TR-87-24.

4. Quentmeyer, R.J., "Experimental Fatigue Life Investigation of Cylindrical Thrust Chambers," NASA TMX-73665.

5. White, F.M. (1974) Viscous Fluid Flow, McGraw-Hill Book Company, New York.

6. Baldwin, B.S., and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows." AIAA Paper No. 78-257, AIAA 16th Aerospace Sciences Meeting, January 1978.

7. Kadle, D.S., and Sparrow, E.M., "Numerical and Experimental Heat Transfer and Fluid Flow in Longitudinal Fin Arrays," Journal of Heat Transfer, February 1986, Volume 108, p. 16-23.

8. Incropera, F.P. (1981) Fundamentals of Heat Transfer, John Wiley and Sons, New York.

NASA
National Aeronautics and Space Administration

Report Documentation Page

1. Report No. NASA TM-103293	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle A Three-Dimensional Turbulent Heat Transfer Analysis for Advanced Tubular Rocket Thrust Chambers		5. Report Date
		6. Performing Organization Code
7. Author(s) Kenneth J. Kacynski		8. Performing Organization Report No. E-5753
		10. Work Unit No. 591-41-21
9. Performing Organization Name and Address National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135-3191		11. Contract or Grant No.
		13. Type of Report and Period Covered Technical Memorandum
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, D.C. 20546-0001		14. Sponsoring Agency Code

15. Supplementary Notes

Prepared for the JANNAF Propulsion Conference, Anaheim, California, October 3-5, 1990.

16. Abstract

Heat transfer was analyzed in the throat region of a plug and spool rocket engine for both smooth and corrugated walls. A three-dimensional, Navier-Stokes code was used for the analysis. The turbulence model in the code was modified to handle turbulence suppression in the crevice region of the corrugated wall. Circumferential variations in the wall heat transfer were predicted for the corrugated wall. The overall heat transfer at the throat for the corrugated wall was 34 percent higher than it was for the smooth wall for comparable rocket flow conditions.

17. Key Words (Suggested by Author(s)) Tubular thrust chamber; Three-dimensional heat transfer; Corrugated wall; Plug-spool thrust chamber	18. Distribution Statement Unclassified – Unlimited; Subject Category 34

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of pages	22. Price*
Unclassified	Unclassified	7	A02

*For sale by the National Technical Information Service, Springfield, Virginia 22161